

Fundamentals of GD&T 1-Day Overview Workshop (ASME Y14.5M-1994)			
Session	Topic	Topics	Exercise
1		Course Goals and Introduction	
	1	Comparison Between Geometric Tolerancing and Coordinate Dimensioning: Coordinate tolerancing, geometric tolerancing, benefits of GD&T, ASME Y14.5M-1994	1
	2	GD&T Terminology: Eight common terms	2
	3	Symbols and Modifiers: GD&T symbols and modifiers	3
	4	GD&T Rules and Major Concepts: Rules #1 & 2; basic dimensions; virtual condition; inner, outer, and worst-case boundary; bonus	4
	5	Form Controls: Flatness, straightness, circularity, cylindricity	5
LUNCH			
2	6	The Datum System: Planar datums, size datums, datum targets	6
	7	Orientation Controls: Perpendicularity, angularity, parallelism	7
	8	Location Controls: Position	8
	9	Runout Controls: Circular and total runout	9
	10	Profile Controls: Profile of a surface	10
		Summary/Questions	

